

Simulations of flow around a three-dimensional square cylinder using LES and DNS

Tufan Arslan^{*}, George K. El Khoury^a, Bjørnar Pettersen^a and Helge I. Andersson^b

^{*}Department of Marine Technology
Norwegian University of Science and Technology, 7491 Trondheim, Norway
e-mail: tufan.arslan@ntnu.no,

^aDepartment of Marine Technology
Norwegian University of Science and Technology, 7491 Trondheim, Norway
e-mails: george.e.khoury@ntnu.no, bjornar.pettersen@ntnu.no

^bDepartment of Energy and Process Engineering
Norwegian University of Science and Technology, 7491 Trondheim, Norway
e-mails: helge.i.andersson@ntnu.no

Abstract. *The aim of the present study is to investigate the turbulent flow around a square cylinder. This is one of the most common studied bluff bodies. The unsteady flow is studied by using computational fluid dynamics (CFD) tools and different turbulence models. Simulation results are compared to laser doppler velocimetry (LDV) measurement data of the flow field published in the literature. The results are also compared with other numerical studies which can be used as verification. In addition, computed force coefficients are compared with experimental measurements.*

Keywords: LES, DNS, bluff body, numerical simulation, square.

1 INTRODUCTION

The flow past a square cylinder has been studied by many investigators for a long period of time. Large eddy simulation (LES) is a good approach for calculation of such unsteady turbulent flows which are three-dimensional and have strong separation and vortex shedding features. The flow past a square cylinder is often used as a benchmark case for testing various sub-grid scale models of LES, as in Sohankar et al. [1] and Rodi et al. [2]. The numerical results in the literature are comprehensive and it is often compared to the experimental works by Lyn et al. [3] and Durao et al. [4]. In more recent years Brun et al. [5] investigated the coherent structures in the flow around the square cylinder both numerically and experimentally. Fröhlich and Terzi [6] coupled LES and Reynolds-Averaged Navier-Stokes(RANS) to simulate the same flow. Filippini et al.[7] and Grigoriadis et al.[8] have investigated the flow around the square cylinder with LES model, Wissink [9], Djedini and Antonia[10] performed DNS.

The codes, FLUENT, MGLET and OpenFOAM have been used in present work for numerical simulations. These codes use the finite volume method to solve the Navier-Stokes equations. FLUENT (FL) is a commercial CFD software developed by ANSYS whereas OpenFOAM (OF) is an open-source code. MGLET is an in-house CFD code by Manhart [15] used for DNS.

2 FLOW CONFIGURATION AND COMPUTER EXPERIMENTS

2.1 Flow Model and Numerical Solution Procedure

In the present work, two different sub-grid scale models are used for LES simulations, i.e., the Smagorinsky's eddy-viscosity model (SM) and the kinetic energy transport model (KET). Simulation results from these two different models are compared with experimental results. For LES, two different Reynolds numbers have been studied, i.e Re=14000 and 21400. For DNS only the Re=14000 case is considered, The Reynolds number is based on the free-stream velocity (U_0) and the crosssection length of the square cross-section.

The governing equations solved in LES modeling are a set of filtered Navier-Stokes (N-S) equations. The continuity and filtered N-S equations can be written as

$$\frac{\delta \bar{u}_i}{\delta x_i} = 0 \quad (1)$$

and

$$\frac{\delta}{\delta t}(\bar{u}_i) + \frac{\delta}{\delta x_j}(\bar{u}_i \bar{u}_j) = -\frac{\delta \bar{P}}{\delta x_i} - \frac{\delta \tau_{ij}}{\delta x_j} \frac{\delta}{\delta x_j} \left(\nu \frac{\delta \bar{u}_i}{\delta x_j} \right) \quad (2)$$

where the overbar points out the filtering operation. τ_{ij} is the subgrid-scale turbulent stress defined by

$$\tau_{ij} \equiv (\bar{u_i u_j} - \bar{u_i} \bar{u_j}) \quad (3)$$

and will to be modeled using the Smagorinsky model (SM). In the original eddy-viscosity model proposed by Smagorinsky [11], the subgrid-scale stresses are modeled using

$$\tau_{ij} - \frac{1}{3} \delta_{ij} \tau_{kk} = -2C_s \overline{\Delta}^2 |\overline{S}| \overline{S}_{ij} \quad (4)$$

where C_s is the model coefficient ($C_s = 0.1 \sim 0.2$), S_{ij} is the resolved rate-of-strain tensor, $|\overline{S}| = \sqrt{2\overline{S}_{ij}\overline{S}_{ij}}$ is its modulus and $\overline{\Delta} \equiv V^{1/3}$ where V is the volume of the cell. In the dynamic procedure, the model coefficient C_s is computed during the LES simulation (DSM) on-the-fly. Details of the procedure can be found in Germano et al. [12].

In the kinetic energy transport (KET) model, the unresolved stresses are modeled by solving a transport equation for the kinetic energy. More details can be found in Kim and Menon [13].

LESs are performed by the FL and OF software. FL employs a cell-centered finite-volume method that permits use of polyhedral cells. Higher order methods are used for discretization of the N-S equations. In order to ensure accuracy, efficiency and stability, bounded central-differencing is adopted to solve filtered momentum equations. This is a non-dissipative, second-order-accurate scheme combined with a nonlinear flux-limiter to suppress numerical oscillations. For pressure-velocity coupling, the PISO algorithm is used. The temporal discretization in the segregated solvers employs a fully implicit, three-level, second-order accurate scheme. The discretized equation system is solved using the point-wise Gauss-Seidel iterative algorithm with and acceleration of an algebraic multi-grid (AMG) method. More details can be found in Kim [14].

DNS calculations have been performed by the direct numerical simulation solver MGLET [15]. MGLET is a finite-volume code in which the Navier-Stokes equations are discretized on a staggered Cartesian mesh with non-equidistant grid-spacing. The discretization of advection and diffusion terms is second-order accurate. For time integration an explicit third-order Runge-Kutta scheme is used. The Poisson equation for the pressure is solved by an implicit procedure [16].

In the OpenFOAM (OF) finite volume open source code, the cell-centre values of the variables are interpolated at face locations using second-order central difference scheme for the diffusive terms. The convection terms are discretized by the so-called limited linear scheme, a second order accurate bounded total variational diminishing (TVD) scheme resulting from the application of the limiter (see Sweby[17]) to the central differencing. The advancement in time is obtained by the implicit two-step second order backward differentiation formula. The pressure-velocity coupling is applied via the pressure-implicit PISO algorithm, using predictor-corrector approach. The discretized equation system is solved by Preconditioned Conjugate Gradient (PCG) methods, see Jakobs [18].

2.2 Computational Domain, Grid and Boundary Conditions

Fig. (1) shows the the computational domain and its boundaries. The incoming flow velocity U_0 corresponds to Reynolds number 14000 and 21400 ($Re = U_0 D / \nu$) where D is the cross-section length of the square cylinder. The turbulence intensity of the incoming flow is 2% for $Re = 21400$ and 6% for $Re = 14000$ matching the turbulence level in the experimental circula-

tion tunnel. The outlet surface has zero pressure condition (atmospheric) as seen in Fig. (1). Simulations are performed with periodic conditions at the side walls. The cylinder surface is considered as no-slip wall. The upper and lower surfaces are defined as symmetry conditions.

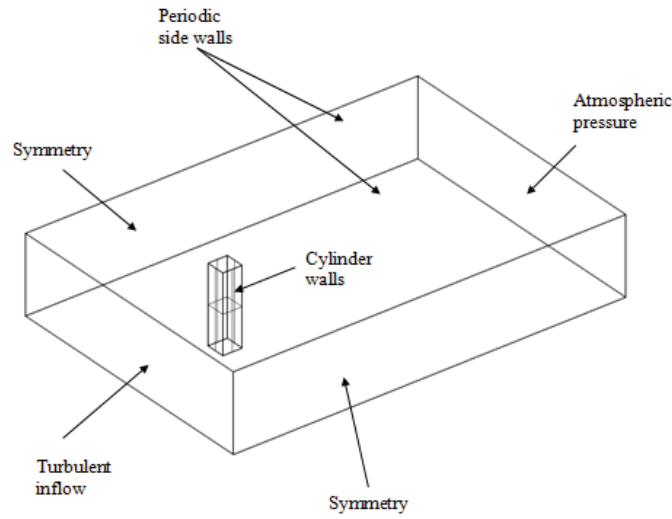


Figure 1. 3D Computational domain with boundary surfaces and its boundary conditions.

The size of the entire 3D computational domain can be seen in Fig. (2). The inlet and outlet are a distance $4.5D$ and $15D$ away from the rectangular section's front and back surfaces, respectively. The length of the cylinder in spanwise direction is $4D$. The distance from the cylinder bottom and upper surface to the boundary surfaces of computational domain is $6.5D$. The size of the computational domain is comparable with works reported in the literature [1], [2], [5], [7], [8].

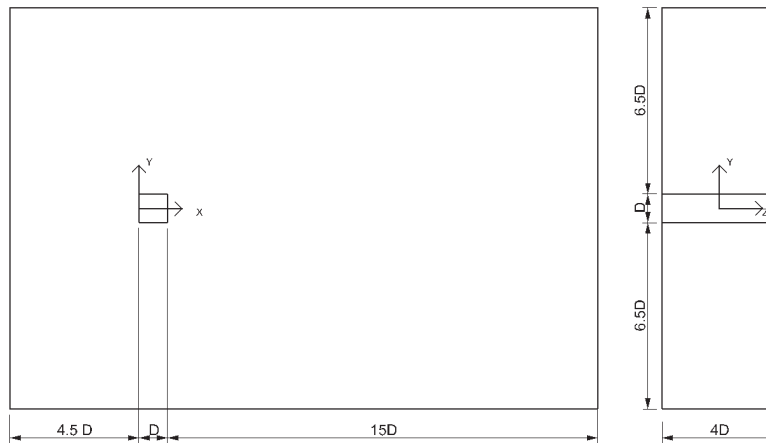


Figure 2. Size of the 3D computational domain.

In the LES calculations, a hexagonal type grid with unstructured quadrilateral type elements in streamwise and vertical direction (xy plane) is used as seen in Fig. (3a) and Fig. (3c). The grid is structured in spanwise direction (z direction). Close to the square cylinder wall, structured grid layers are employed in order to have sufficient small mesh size to resolve the

boundary layer (Fig. 3b.) The number of structured cells in these layers is 15 for the cylinder surface. The grid spacing normal to the wall for the first layer is $0.002D$ at the cylinder and expands out from the surface with the ratio equal to 1.2. Total number of elements in the 3D domain is varying from 1 to 4 million dependent on the expansion ratio from cylinder surface and resolution in the wake. Outside the hexahedral layer the quadrilateral elements in xy plane expand with the ratio 1.1 and reach the maximum size of $0.2D$ near the outer boundaries. This expansion is controlled in the wake region close to the rear of the cylinder (see Fig. 3a). The grid size is kept between $0.025D$, $0.02D$ and $0.015D$ in this area for the meshes which has 1 million, 2 million and 4 million total elements respectively as seen in Table 1..The number of elements for discretizing the rectangular section with clustering towards the corners and element size in spanwise direction are showed in Table 1. The resulting y^+ values are in the order of 1 at the cylinder surface.

	Total number of elements	Number of elements in spanwise direction	Element size in wake region	Number of elements on the cylinder surface (crosssection)
MESH1	1 million	30	$0.025D$	110×110
MESH2	2 milion	32	$0.02D$	120×120
MESH3	4 million	39	$0.15D$	140×140

Table 1: Computational grid resolution for LES calculations.

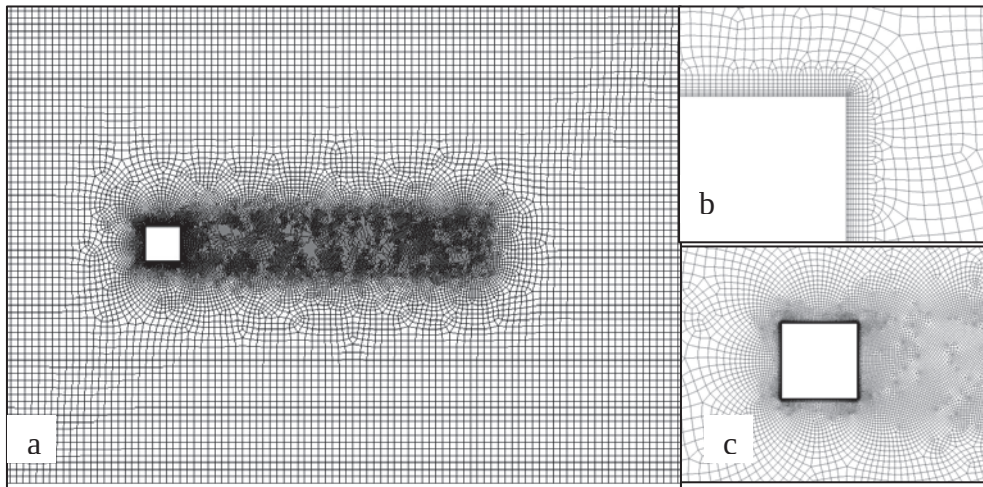


Figure 3. Cross-section of the 3D computational mesh, whole domain (a), around the trailing upper corner (b), around the square cylinder (c).

For DNS simulations, a structured mesh ($850 \times 450 \times 400$ elements) is used with the same domain size. In order to verify the current simulation is an sufficiently resolved direct numerical simulation, the Kolmogorov length scale η is estimated as $\varepsilon = (\nu^3 / \varepsilon)^{1/4}$, where the energy dissipation rate $\varepsilon = \nu (\partial u_i / \partial x_k)^2$ is calculated from the simulation flow field. The local grid size Δ relative to Kolmogorov length scale is presented in Table 2 at different streamwise positions in the mid-section. The data in Table 2 shows that the adopted grid spacing is sufficiently fine to resolve almost all essential scales in the wake ($\Delta \sim < 7\eta$). This value is slightly higher but comparable with results from literature, Khoury et.al. [19].

	X/D=1.5	X/D =2.5	X/D =4	X/D =9
Y/D=0.5	6.22	6.69	7.71	8.56
Y/D=0.375	6.04	6.84	8.11	8.97
Y/D=0.25	5.78	7.03	8.31	9.39
Y/D=0.125	5.71	7.20	8.36	9.11
Y/D=0	5.08	7.37	8.17	9.45

Table 2: Grid resolution $\Delta = \max(\Delta x, \Delta y, \Delta z)/\eta$ at various streamwise locations (Re=14000).

The time is non-dimensionalized by the uniform inlet velocity (U_0) and crossection length D as $T=tU_0/D$ and the simulations are carried out for $T=140$ in order to reach statistical convergence. The total simulation time corresponds to approximately 20 vortex shedding cycles. The Strouhal number, $St = fD/U_0$ where f is the frequency, is gathered from the lift coefficient as a result of a FFT analysis. Mean drag and deviation of lift coefficients are calculated from the instantaneous forces.

3 RESULTS

In the LES calculations, at Reynolds number $Re=21400$, the effect of the temporal and spatial discretization on the simulation results is investigated by applying three different mesh sizes with different time steps. Beside this, the effect of the inlet turbulence level on the flow field has also been investigated.

Case No.	mesh size (million)	time-step (seconds)	sub-grid scale	inlet turb.	Solver	$\overline{C_D}$	C_L'	St
CASE-1	1	0.0004	SM	2 %	OF	2.15	1.37	0.135
CASE-2	1	0.00025	SM	2 %	OF	2.16	1.37	0.133
CASE-3	1	0.0001	SM	2 %	OF	2.15	1.45	0.137
CASE-4	2	0.00025	SM	2 %	OF	2.13	1.39	0.133
CASE-5	4	0.00025	SM	2 %	OF	2.13	1.36	0.133
CASE-6	1	0.00025	KET	2 %	OF	2.15	1.31	0.133
CASE-7	1	0.00025	SM	no turb.	OF	2.12	1.21	0.134
CASE-8	2	0.00025	SM	2 %	FL	2.21	1.4	0.133
CASE-9	1	0.00025	DSM	2 %	FL	2.21	1.45	0.073
CASE-10	1	0.00025	SM	2 %	FL	2.25	1.43	0.127
Exp. [3]						2.1		0.132
LES [1]	1	0.025	OEDSMF			2.09		0.132

Table 3: Mean drag and RMS of lift coefficients and Strouhal numbers (Re=21400).

The calculated lift and drag force coefficients in addition to the Strouhal numbers (based on lift coefficients) are compared with experimental data. In Table 3., the calculated mean drag coefficients, standard deviation of lift coefficients and Strouhal numbers are tabulated for 1, 2 and 4 million elements and different time steps (0.0001, 0.00025 and 0.0004 sec.). Use of SM, DSM and KET sub-grid scale models are also shown. In CASE-7, the incoming flow is considered smooth in order to investigate the effect of inlet turbulence (CASE-2). The re-

sults from FLUENT and OpenFOAM calculations are also compared. (CASE-2 vs. CASE-10 and CASE-4 vs. CASE-8)

For LES calculations at Reynolds number $Re=14000$, the computational mesh has one million elements (MESH1) and the time step is 0.00025 sec. The time and spanwise averaged streamwise velocity obtained by OpenFOAM using Smagorinsky sub-grid scale model is shown in Fig. 4 together with the DNS and experimental results. Here, U_0 is the free stream velocity and D is the length of the square cylinder cross-section.

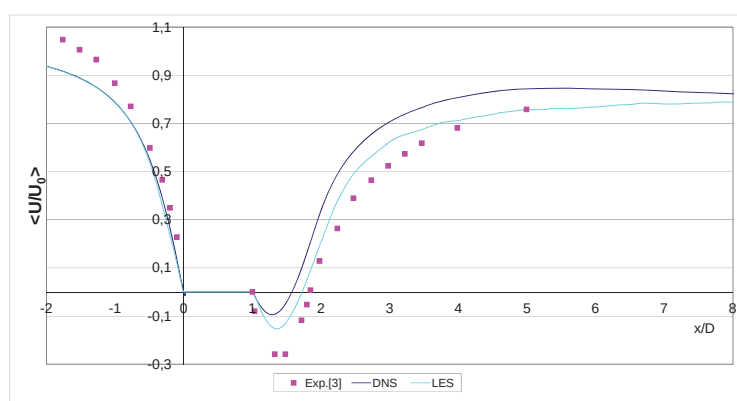


Figure 4. Time and spanwise averaged streamwise velocities at centerline in front and behind the square cylinder ($Re=14000$, MESH1).

As seen in Fig. (4), in LES, the minimum of the mean velocity in the near wake is estimated better than by DNS. This region is close to the center of the main vortex and kinetic energy has its maximum value. The fair prediction of the mean velocity may be due to under-estimation of the velocity fluctuations because of slightly higher Δ values and the effect of unresolved small turbulent eddies in the wake region. However, as seen in Fig. (5), the time and spanwise averaged Reynolds normal stresses $\langle u'u' \rangle$ agree well with experimental results for both DNS and LES. But the vertical component of Reynolds stresses $\langle v'v' \rangle$ has been under-predicted in the near wake, 1.5D from origin of the coordinate system which is shown in Fig. (2). At the section 2.5D from origin, the $\langle v'v' \rangle$ values have been predicted well for both DNS and LES but here $\langle u'u' \rangle$ values are different from experimental results. Consequently, in the wake, Reynolds stresses have not been predicted well at the same time for both components. This results in the difference between calculations and experiments for the turbulent kinetic energy in the wake.

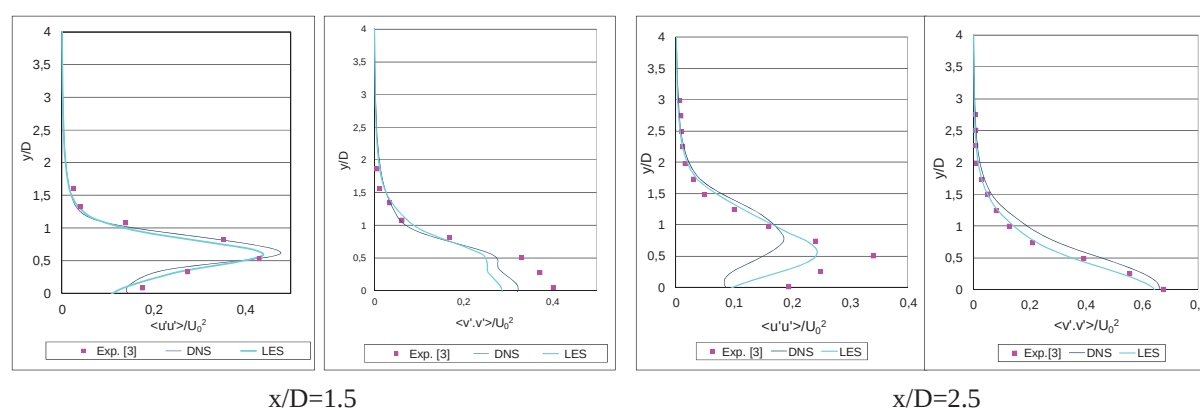


Figure 5. Time and spanwise averaged Reynolds normal turbulent stresses in the wake, $Re=14000$.

In Fig. (6), the measured (Duraio et.al.[4]) and calculated turbulent kinetic energy contours by LES and DNS are plotted. The distribution of the turbulent kinetic energy, k , is calculated on the basis of local isotropy, i.e. $k = 3/4(\overline{u'^2} + \overline{v'^2})$. The contours reveal distributions which are qualitatively similar to each other in the simulations and the experiment. The shear layer surrounding the recirculation bubble has intense velocity fluctuations and turbulent kinetic energy increase as the downstream distance increase, with maximum values around the center of the main circulation bubble.

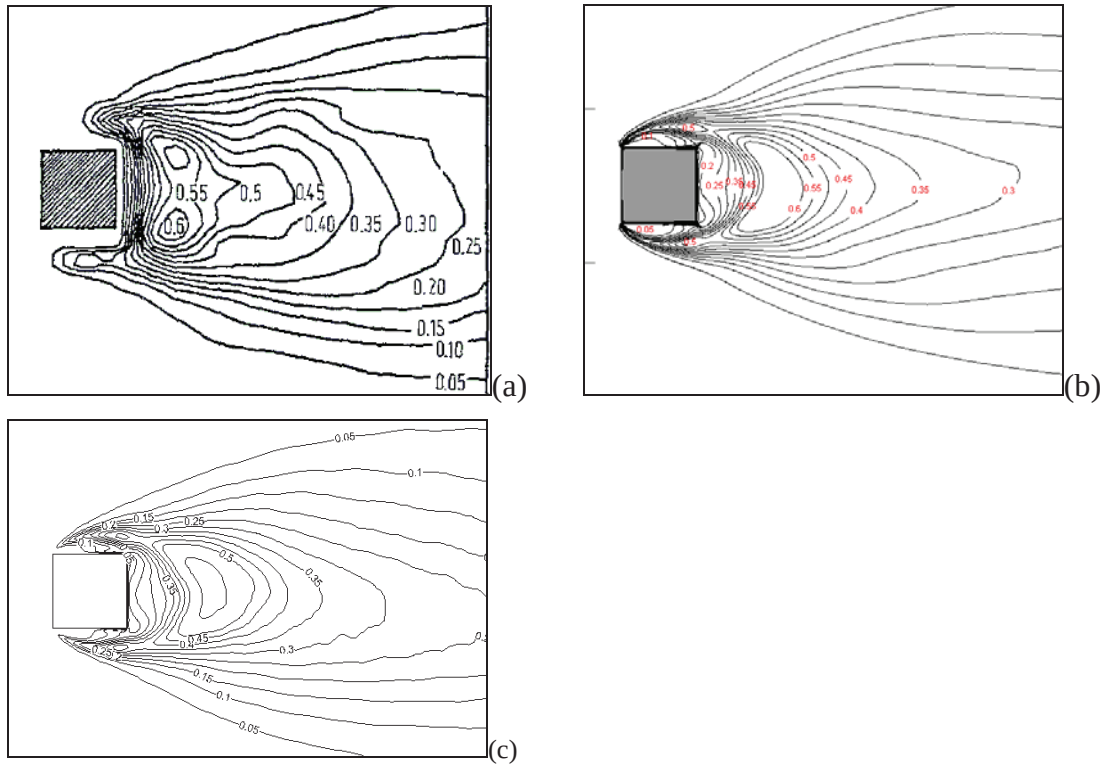


Figure 6 Spanwise averaged and normalized kinetic energy contours, (a) experiment [4] , (b) DNS and (c) LES at $Re=14000$.

In Fig. (7b), are shown experimental visualization results by Duraio et.al[4] for Reynolds number $Re=14000$. Calculations of the instantaneous streamwise velocity contours have been performed by MGLET with DNS, and are shown in Fig (6 a). The streamwise velocity is normalized with free stream velocity (U_0) and spanwise averaged. Flow visualization has been made by injection of a solution of fluorescein-sodium at the upstream face of the cylinder. The calculated shear layer and vortex formation are very close to the experimental visualization as seen in Fig. (6)

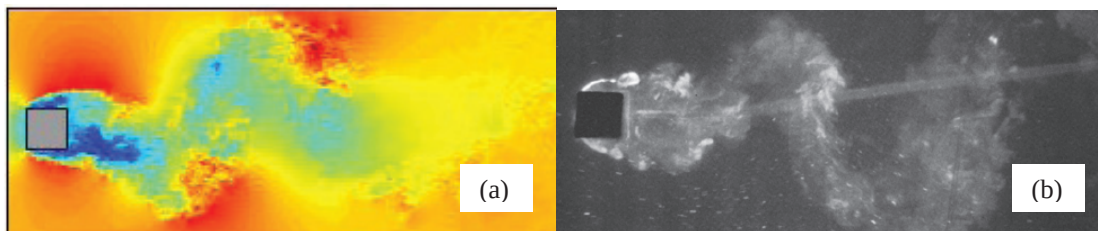


Figure 7. Instantaneous normalized streamwise velocity (U/U_0) contours by (a) DNS and (b) flow visualization experiments [4] for $Re=14000$.

4 CONCLUSIONS

- For Reynolds number $Re=21400$, the results from LES simulations with both codes and sub-grid scale models give good agreement with experimental data. One million elements and 0.0004 sec. time step size seem enough to predict acting forces and frequency of the lift with reasonable accuracy for the OF code. However the FL code gives poor results for the Strouhal number, and the mean drag coefficient is slightly over-predicted for coarse meshes (see CASE9 and CASE-10 in Table 3). The OF code seems less sensitive to grid resolution.
- Simulations show that the lift coefficient is sensitive to inlet turbulence level.
- For Reynolds number $Re=14000$, the variation of the streamwise component of the velocity is well predicted while the vertical component is underpredicted near the centerline in the wake ($y/D < 0.5$) for both LES and DNS compared to experimental results. The calculated mean streamwise velocity is also underpredicted compared to experiments in the near wake and probably inside the separation bubble ($x/D < 2$).
- DSM and KET which are more costly models, do not bring significant advantages for predicting drag and lift coefficients. The SM model can be used to simulate such flows.

ACKNOWLEDGEMENTS

Ph.D. research fellowship for the first author was provided by the Research Council of Norway. Computing time was granted by Research Council of Norway via NOTUR (The Norwegian Metacenter for Computational Science).

REFERENCES

- [1] A. Sohankar, L. Davidson and C. Norberg. Large eddy simulation of flow past a square cylinder: comparison of different subgrid scale models. *Journal of Fluids Engineering*, **122**, 39–47, 2000.
- [2] W. Rodi, J. H. Ferziger, M. Breuer and M. Pourquié. Status of large eddy simulation: Results of a workshop. *Journal of Fluid Engineering*, **119**, 248-262, 1997.
- [3] D. A. Lyn, S. Einav, W. Rodi and J. H. Park. A Laser-doppler velocimetry study of the ensemble-averaged characteristics of the turbulent near wake of a square cylinder. *Journal of Fluid Mechanics*, **304**, 285–319, 1995.
- [4] D. F. G. Durao, M. V. Heitor and J. C. F. Pereira. Measurements of turbulent and periodic flows around a square cross-section cylinder. *Experiments in Fluids*, **1**, No. 5, 298-304, 1988.
- [5] C. Brun, S. Aubrun, T. Goossens and P. Ravier. Coherent structures and their frequency signature in separated shear layer on the sides of a square cylinder. *Flow Turbulence and Combustion*, **81**, 97-117, 2008.
- [6] J. Fröhlich and D. V. Terzi. Hybrid LES/RANS methods for the simulations of turbulent flows. *Progress in Aerospace Sciences*, **43**, 349–377, 2008.
- [7] G. Filippini, G. Franck, N. Nigro, M. Sorti and J. D’elia. Large eddy simulations of the flow around a square cylinder, *Mecánica Computacional, Volume XXIV*, Buenos Aires, Argentina, November 2005.

- [8] D. G. E. Grigoriadis, J. G. Bartzis and A. Goulas. LES of the flow past a rectangular cylinder using the immersed boundary concept, *International Journal for Numerical Methods in Fluids*, **41**, 615-632, 2003.
- [9] D. Wissink. DNS of 2D turbulent flow around a square cylinder. *International Journal for Numerical Methods in Fluids*, **25**, 51-62, 1997.
- [10] L. Djedini and R. A. Antonia. Shear layer instability in a low Reynolds number wake, *Proceedings of 8th International Symposium on Engineering Turbulence Modelling and Measurements*, ETMM8, ERCOFTAC, Marseille, France, 9-11 June 2010.
- [11] J. Smagorinsky. General circulation experiments with the primitive equations. I. The basic experiment. *Monthly Weather Review*, **91**, 99–164, 1963.
- [12] M. Germano, U. Piomelli, P. Moin and W. Cabot. A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids*, **3**, 1760-1765, 1991.
- [13] W. W. Kim and S. Menon. Application of the localized dynamic subgrid-scale model to turbulent wall-bounded flows. *Technical Report AIAA-97-0210*, 35th Aerospace Sciences Meeting, American Institute of Aeronautics and Astronautics, Reno, NV, January 6-9, 1997.
- [14] S. -E. Kim. Large eddy simulation using unstructured meshes and dynamic subgrid-scale turbulence models. *Technical Report AIAA-2004-2548*, 34th Fluid Dynamics Conference and Exhibit, American Institute of Aeronautics and Astronautics, Portland, Oregon, June 28-July 1, 2004.
- [15] M. Manhart. A zonal algorithm for DNS of turbulent boundary layers. *Computers and Fluids*, **33**, 435-461, 2004.
- [16] H. Stone. Iterative solutions of implicit approximations of multidimensional partial differential equations. *SIAM, Journal of Numerical Analysis*, **5**, 530-538, 1968.
- [17] P. K. Sweby. High resolution schemes using flux limiter for hyperbolic conservation laws. *SIAM, Journal of Numerical Analysis*, **21**, 995-1011, 1984.
- [18] D. A. H. Jakobs. Preconditioned Conjugate Gradient methods for solving systems of algebraic equations. *Central Electricity Research Laboratories Report*, RD/L/N193/80, 1980.
- [19] G. K. E. Khoury, H. I. Andersson and B. Pettersen. Crossflow past a prolate spheroid at Reynolds number of 10 000. *Journal of Fluid Mechanics*, **659**, 365–374, 2010.